AR-010-220

DSTO-TR-0529

Implementation of a Unified Constitutive Model into the PAFEC Finite Element Package: Final Report

J. Paul

APPROVED FOR PUBLIC RELEASE

DEPARTMENT OF DEFENCE

DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION

Implementation of a Unified Constitutive Model into the PAFEC Finite Element Package: Final Report

J. Paul

Airframes and Engines Division
Aeronautical and Maritime Research Laboratory

DSTO-TR-0529

ABSTRACT

A unified constitutive material model was implemented in to the PAFEC Finite Element Package Level 8.1. The modifications allow the PAFEC Package to perform time dependent plasticity. This includes the calculation of the updated material Jacobian used in the formulation of the global stiffness matrix. Automated time stepping, reduced storage requirements and structural convergence features were also included in the code. A series of test analyses addressing different aspects such as element types and loading conditions were performed and shown to agree with expected results.

RELEASE LIMITATION

Approved for public release

DEPARTMENT OF DEFENCE

DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION

Published by

DSTO Aeronautical and Maritime Research Laboratory
PO Box 4331
Melbourne Victoria 3001

Telephone: (03) 9626 7000
Fax: (03) 9626 7999

AR-010-220
May 1997

APPROVED FOR PUBLIC RELEASE

Implementation of a Unified Constitutive Model into the PAFEC Finite Element Package: Final Report

Executive Summary

The conditions under which the RAAF operate the F-111 aircraft are conducive to cracking in the structurally critical wing pivot fitting, and certification testing in the Cold Proof Load Test (CPLT) has demonstrated failure in the same region. The main area of concern is the Fuel Flow Vent Hole Number 13 (FFVH#13), in the wing pivot fitting. In order to calculate inspection intervals, a detailed knowledge of the elastic and residual stress fields is required for these locations.

Classical plasticity solution techniques reveal severe limitations in representing material behaviour under non-symmetric cyclic loading. As a result AMRL researched and developed an alternative constitutive material model which would provide an improved representation of the residual stress field after a CPLT type load cycle. This report describes the modifications made to the PAFEC Finite Element Package in order to implement the unified constitutive material model, and shows that it produces validated results.

The solution capability provided by the implementation of the unified constitutive material model into PAFEC will be used extensively in the F-111 Structural Integrity Task to perform the detailed plasticity analyses on the F-111 FFVH#13. The elastic and residual stress fields developed will then be used as an important ingredient for the determination of the inspection interval for this critical location.

Authors

J. Paul
Airframes and Engines Division

Mr Paul has a degree in Aeronautical Engineering and a Masters in Mechanical Engineering in the repair of thick composite structures. He has worked in the finite element field for 13 years providing AMRL with a high level of expertise in the area of computational analysis which has been utilised to solve a variety of RAAF related stress/strain problems seen on the F-111 and F/A-18 aircraft. He is currently the Functional Head of the Computational Stress Analysis Facilities within AED and leads the team working on the F-111 Structural Integrity Task, which provides the residual stress input required for the calculation of the inspection interval for the Fuel Flow Vent Hole #13 location in the F-111 Wing Pivot Fitting.

Contents

1. Introduction

Classical techniques of modelling material creep and plasticity have acceptable accuracy provided the inelastic strains are kept small. However, they are poor in describing material behaviour where significant inelastic strains, followed by unloading and cyclic re-loading are involved. Several structurally significant problems on the F-111C aircraft involve large cyclic plasticity effects. The Cold Proof Load Test's (CPLT) non-symmetric cyclic loading induces large residual stresses into the structure and these have a substantial effect on fatigue life. This accordingly influences the Royal Australian Air Force's (RAAF) structural maintenance policies.

To overcome the limitations of conventional material models, new unified constitutive models have over the years been developed by researchers. In 1988 the Aeronautical and Maritime Research Laboratory (AMRL) examined several of the material models described in a NASA report [1]. The model chosen to tackle large cyclic plasticity effects was developed by Prof. D.C. Stouffer and V. S. Battachar at the University of Cincinatti, Ohio, USA. Original work at AMRL utilised a one-dimensional FORTRAN program for this model developed by Prof. D.C. Stouffer and colleagues, reviewed in Reference 2.

In 1990 the first version of the PAFEC code implementation of a unified constitutive material model was completed [3]. Reference 3 describes the modifications that were required to be made to the PAFEC Finite Element Program. The initial implementation resulted in very small time steps and required large disk storage areas as a result of the numerical integration scheme employed. In addition the global stiffness matrix was not updated at each iteration which reduced structural convergence rates. Recently development work addressing these deficiencies via the solution of the constitutive differential equations and the material Jacobian determination has been performed by Trippit and Jones [4,5]. In 1995, AMRL contracted Monash University to provide details of the solution technique and material Jacobian determination and effect the installation of these improvements into AMRL's version of the PAFEC code [9].

This report describes the modifications made to the PAFEC Finite Element Program to incorporate the new unified constitutive material model. Updated input data definitions (PAFEC User Modules) are described in detail in order to allow users to apply the constitutive model in their analyses.

2. The Unified Constitutive Model

Unified constitutive models have advantages over classical techniques which allow improved representation of strain rate and cyclic response effects [6]. These effects allow the calculation of time dependant variables such as strain, creep, stress relaxation and cyclic hardening/softening.

The unified constitutive model implemented into PAFEC FE was developed initially by Ramaswamy, Stouffer and Bodner [1] and further developed by Trippit et al [4 and 5] and Searl and Paul [7]. It consists of the inelastic flow equation and the rate forms of the state variables: back stress Ω_{ij} and drag stress Z. In this formulation the inelastic flow equation is written as:

$$\dot{\varepsilon}^{I}_{ij} = D\exp\left[-\frac{1}{2}\left(\frac{Z^2}{3K_2}\right)^{n}\right]\frac{\left(S_{ij}-\Omega_{ij}\right)}{\sqrt{K_2}} \tag{1}$$

where $\dot{\varepsilon}^{I}$ is the inelastic strain rate, K_2 is the second invariant of the deviatoric over-stress tensor, $S_{ij}-\Omega_{ij}$, and D and n are temperature and strain rate dependent material parameters. Here the evolution equation for the back stress Ω_{ij} is defined as:

$$\Omega_{ij} = f_2 S_{ij} + \Omega^{I}_{ij} \tag{2}$$

where f_2 is a material. The inelastic back stress Ω^{I}_{ij} is calculated by integrating the following,

$$\dot{\Omega}^{I}_{ij} = f_1\,\dot{\varepsilon}^{I}_{eff} - \frac{3}{2}f_1\,\frac{\Omega_{ij}}{\Omega_{max}}\,\dot{\varepsilon}^{I}_{eff} \tag{3}$$

where $\dot{\Omega}^{I}_{ij}$ is the inelastic back stress rate, Ω_{max} is the maximum value of back stress expected, $\dot{\varepsilon}^{I}_{eff}$ is the effective inelastic strain rate and f_1 is a hardness related material parameter.

The drag stress Z is given by,

$$Z = Z_1 + (Z_0 - Z_1)\exp\left(-m\,\varepsilon^{I}_{eff}\right) \tag{4}$$

where Z_0 is the initial value of drag stress, Z_1 is the final value of drag stress and m defines the rate at which Z evolves from Z_0 to Z_1. The effective inelastic strain ε^{I}_{eff}, is given by:

$$\varepsilon^{I}_{eff} = \sqrt{\frac{2}{3}\varepsilon^{I}_{ij}\varepsilon^{I}_{ij}} \tag{5}$$

where ε^{I}_{ij} is the inelastic strain tensor.

Details of the derivation of the material parameters D, n, f_1, f_2, Ω_{max}, Z_0, Z_1 and m for D6ac steel can be found in Reference 7.

3. Existing PAFEC Plasticity Algorithm

To solve non-linear plasticity problems the current PAFEC plasticity algorithm uses successive incremental application of loads with an iterative approach using the Newton-Raphson scheme to obtain convergence. The material is assumed to behave elastically before yield, according to Hooke's law. Yielding is deemed to have occurred when the effective stress of a Gauss point traverses the boundary of the yield function.

There are currently three yield criteria available: 1) the Von Mises criterion, 2) the Drucker-Prager criterion and 3) a user defined criterion. PAFEC's constitutive equations use the Prandtl-Reuss relationships, and optional hardening rules can be selected ranging from pure isotropic hardening to pure kinematic hardening. The user is allowed to choose between the Euler or Runge-Kutta-England integration schemes. At the onset of plastic yielding the algorithm performs a sub-iteration process in order to converge to the best estimate of the stress, and then proceeds to check for structural convergence via the residual forces (several different convergence criteria are available). If structural convergence has not been achieved then the program will iterate again with modified increments in strains. These modified strains are estimates based upon the balance between the internal and external work and are calculated from the residual force vectors. Once convergence is achieved the program moves to the next load increment.

The main controlling FORTRAN routines are shown in Figure 1. A brief description of the function of each routine is also included in this figure and the detailed algorithm was previously described in [3].

4. Implementation

4.1 Introduction

The major difference between the new constitutive equations and the standard PAFEC solution is the inclusion of time dependent plasticity which can describe creep effects. Other features such as automatic time step load calculation, an additional numerical solver for the constitutive equations, material Jacobian calculation and structural convergence time stepping control have also been included.

The fundamental change required to PAFEC's original algorithm involves the replacement of the state determination routines and the re-definition of the load increment. The revised FORTRAN subroutine calls referred to in this report are

presented in Table 1 (new routines) and Table 2 (old PAFEC modified routines). An updated program flow chart of the modified areas has been provided in Figure 2.

Specific details of the overall implementation may be found in Section 4 of Reference 3 and are only be repeated here where changes have been made. A description of additional and enhanced features implemented by this work follows.

4.2 Data Input

The PAFEC system allows users to define data modules which enable easy input of data into the program. With the unified constitutive model implementation, four additional data modules were required. The new data modules are:

1. AUTOMATIC.CONTROL
2. CONSTANTS.NEW.FLOW.LAW
3. UNIFIED.CONTROL
4. TOTAL.TIME.VERSUS.FORCE.DATA

These modules and their influence on the behaviour of the solution are described in Appendix A. The PAFEC modular dictionary has also been modified to include the above modules. Details of these modifications can be found in Appendix B, Section B.1.

4.3 Initialisation

Modifications made to the initialisation routines are detailed below:

4.3.1 Control flags

The LFLAGS common block has been extended in routine *R09702* to include positions 30 to 44. Details of the meaning of each new control flag can be found in Appendix B, Section B.2.

4.3.2 Data validation and initialisation

PAFEC is divided into 10 Phases which perform different functions, see [8]. The PAFEC system generally performs all data validation in Phases 1 to 4, but the new data modules are validated at the initialisation stage of Phase 9. Default values are inserted where applicable if none have been provided, and WARNING messages are printed to the output files. The routine JP1000 performs this function.

4.3.3 Space allocation

In the previous implementation a large number of non-linear data sets were set up to handle the information required to be stored on a Gauss point basis. This has been

made more efficient. Appendix B, Section B.3 contains details of the changed data set configuration with information for the NLSET storage array.

4.3.4 Additional scratch modules

Additional scratch modules are required to hold solution control information, double precision material data, solution recovery information and other data. Details of these modules can be found in Appendix B, Section B.4.

4.4 Multiple Load Case Capability

Code in the previous version was not capable of handling more than one load case. PAFEC does not have this limitation and the routines were re-written allow multiple load cases. The input of total time data via the TOTAL.TIME.VERSUS.FORCE.DATA module (see Appendix A) has been affected by these modifications. This module must now contain the total time data for each individual load case corresponding to the INCREMENTAL module. New routines *JP1105* and *JP1106* have been included to process the Load-Time data and the old routines *JP1100* and *JP1101* have been modified. A negative total time is permitted to indicate an unload condition so that the elastic stiffness matrix should be used at the next load increment.

The capability to have several load cases active at any one time has been incorporated into the modifications, although there is no current requirement. The routines would require validation if this facility is needed in the future.

4.5 Load Step Housekeeping

The majority of the work required to be performed prior to each load step has not been changed in this version, see [3] for specific details of previous changes. However, modifications have been made to allow for the overwriting of the current load increments data. This allows many sub-increments to be performed while storing only selected increments at specified time points. This feature is controlled by the DELTA.STORAGE.TIME.VALUE option in the AUTOMATIC.CONTROL module. The BLOCK.SIZE option allows the program to expand the incremental and dynamic incremental modules beyond the original size defined, thus defining the complete size of the stress/strain output files. Correct use of these two options can limit the size of disk storage requirements for a particular analysis.

The standard increment counter (IBASE(153)) has been replaced with a sub-increment counter (IBASE(180)) that indicates of the total number of increments a solution has performed. The standard increment counter gives the number of increments that have actually been written to the results files.

4.6 Material Jacobian Determination using Perturbation Theory

Reference 9 shows how a material Jacobian is calculated for each Gauss point within the yielding zone, using perturbation theory. The UNIFIED.CONTROL module allows control over the solver method, perturbation parameters and solver tolerances. The routine *kmodel* is the main routine that handles the unified constitutive solution of the equations.

The Jacobian is calculated at the material level and then utilised in the *PL6XXX* series of routines when the tangent stiffness matrix is formulated. Utilising this technique has achieved a dramatic improvement in the number of iterations required to achieve structural convergence within an analysis. The calculation of the Jacobian by perturbation can be quite expensive in terms of CPU time, however this is more than compensated by the improved structural convergence rates. In order to reduce the penalty of calculating the Jacobian, it was been linked to the FREQUENCY and FIX.STIFFNESS options within the CONVERGENCE module, see [8].

4.7 Stress Determination

Three types of elements are currently supported: 1) 3D Isoparametric elements; 2) 2D plane stress and plane strain elements; and 3) Semi-Loof shell elements. Each main routine has been modified to call the main unified constitutive state determination routine J*P3000* which in turn calls *kmodel*. The routine *kmodel* performs the Jacobian perturbation and *kkmodel* performs the state determination for the unified constitutive model. In the previous version *JP3000* called *JP4000* and the *JP4XXX* series of routines has been removed.

Elements that are defined to be elastic bypass the call to *kmodel* and the stress is calculated using the standard techniques. For efficiency, elements defined in the YIELDING.ELEMENTS module [8] are also checked to see if plasticity is occurring and if not are returned to have the elastic stress calculated using the standard techniques.

It should also be noted that the strain definition varies between PAFEC and the constitutive model equations. The inelastic constitutive equations use the tensor definition of strain, but results are output as engineering strains. PAFEC works internally with engineering strains, but outputs its results in terms of tensor strain. This has been taken care of in the routine *JP3000.*

4.8 Livermore Solver

The stress determination routine *kkmodel* utilises the Livermore solver [12] for the ordinary differential equations of the unified constitutive model. This solver is very good when used with stiff and non-stiff systems of differential equations. It has proved to be very successful for strain rate independent materials which cause the

equations to be extremely stiff. The previous implementation, discussed in [9], had to take very small material time steps which resulted in very small structural time steps and large solution times (in the order of weeks real time for the typical F-111C problem). The Livermore solver performs this small stepping internally and very efficiently.

4.9 Post-Convergence Processing

The current modifications allow for the same convergence/non-convergence handling outlined in [3]. The only modification is the time step determination technique which has been re-worked to take full advantage of the large material time steps that are achieved with the use of the Livermore Solver.

4.9.1 Time step determination

In automatic mode, the time step for the next load step is based upon the ability of the material law to maintain integrity as well as how the structure is performing in terms of global convergence.

The unified constitutive routine returns a delta time expansion or contraction factor based upon the level of plasticity at every Gauss point. The original version's routine, *JP3920,* still determines the minimum time step for the structure based upon this factor. Previously this was used for the whole structure, however with the improved solver, the time step tends to be quite large, eg it is not unrealistic to be able to jump from load turning point to load turning point.

The routine *JP1100* has been significantly modified to compute the global time step based upon the following criteria:

1. material convergence,
2. structural convergence,
3. an applied load turning point,
4. turning point recovery (time step after a turning point as been reached),
5. non-convergence situations:
   a) material non-convergence,
   b) maximum iterations exceeded.

The user can tailor structural convergence parameters in the AUTOMATIC. CONTROL module (see Appendix A for specific details).

4.9.2 Phase Output Control

The current time dependent solution algorithm can involve many time steps, and as before there is a need to limit unnecessary I/O in all areas of the program. Since the last report [3], modifications have been made to improve the ability to extract only the required information. This relies heavily on the ability of PAFEC Integrated Graphics System to extract the stress and strain data that is required. Output to the Phase output

file can be controlled by the OUTPUT.PRINT. CONTROL option in the AUTOMATIC. CONTROL module and this is described in Appendix A.

The table output option has not been changed and is still available.

4.9.3 Reduced Integration

In order to attain faster solution times, subroutine *NL0007* was modified so that all available elements, except semi-Loof, utilise reduced integration. This routine has been taken out of the global library and now has to be used separately when required. Using this option can significantly reduce the size of memory (BASE) required to solve large problems.

4.10 Control Option

A control option CONSTITUTIVE has been included into the PAFEC system so that the unified constitutive law can be utilised at any time. This option indicates to the PAFEC system that the unified constitutive object code is to be linked in to the Phase 9 executable.

4.11 Convergence Module

The CONVERGENCE module has some very useful features that can improve the performance of an analysis. Care has been taken to make sure that these options are active and working correctly. Note that the load increment now refers to the stored load increment and not the actual number of increments performed (see Section 4.5).

4.12 Tolerance Module

The TOLERANCE module like the convergence module can be used to tailor and improve the performance of an analysis. This module again uses the stored load increment and not the actual number of increments performed. TOL5 has been generally used, however this tolerance formulation can cause difficulties when trying to converge on a zero applied load value. In these cases the TOL7 (not documented in the PAFEC Level 8.1 manual) has been used successfully.

4.13 PAFEC Modules Disabled

There are two standard PAFEC plasticity modules that are only required so that validation may be passed, but have actually been disabled. They are:

1. STATE.DETERMINATION
2. UNIAXIAL.PROPERTIES

5. System Testing and Validation

5.1 Introduction

The constitutive material model was validated with respect to the PAFEC system to ensure that no array corruption occurred, stresses and strains components were correct and the element stiffness matrix calculation was correctly updated in the global stiffness matrix. Here, the results generated from the test jobs were compared directly with an external FORTRAN program's results for the same set of material parameters. The validation of the unified constitutive model, with respect to D6ac steel can be found in References 7, 10 and 11, where numerical results were directly compared to experimental test data.

5.2 PAFEC Options Validated

5.2.1 Description

Two and three dimensional test cases were generated to demonstrate that the material code was behaving as expected. These test cases included all the element types outlined in Section 5.2.2. Combinations of multiple elements, element types, uniaxial and cyclic loading, plane stress and plane strain, and additional elastic elements were investigated to ensure that the code did not affect or cause corruption to other aspects of the finite element package. The generated solutions were compared against known answers for the particular case being investigated. Table 3 gives a summary of these test jobs with the types of elements and loading conditions which have been validated.

This section presents specific results for two of the validation problems. The first problem was a two dimensional plate (Test 5,Table 3) containing 4 plane strain elements, which was loaded uniaxially at one end and restrained at the other, see Figure 3a and 3b. The structure was displaced at node 3 in the 'y' direction and all nodes (119, 120, 121, 4) in the same plane have their freedoms repeated to provide an even distribution. The applied loading was sufficient to induce a 4% strain in the material. The results for this analysis are shown in Figure 4.

The second validation problem was a three dimension bar (Test 1, Table 5) which is similar to the previous example and contained 8 isoparametric brick elements. The displacement loading was applied at node 5 and all nodes in the constant 'z' plane have their freedoms repeated, see Figure 5a. The structure was restrained on three sides as shown in Figure 5b. Again the loading induced 4% strain and the results are shown in Figure 6.

The simple test models showed that mixing the element types produce valid results. Elements with a local axis system, different from the global axis system, have been shown to produce the correct global stresses.

Simple test problems cannot always trap problems such as array overwriting. A more detailed validation of the complete implementation of the unified constitutive model was performed for a circular and non-circular hole in a plate [10, 11]. Here experimental data was compared to the finite element results and good agreement was achieved.

5.2.2 Supported Element Types

The implementation of the constitutive material model supports the following elements:

1. 3D isoparametric brick and prism elements:
   a) 37100 (8 noded brick)
   b) 37110 (20 noded brick)
   c) 37200 (6 noded prism)
   d) 37210 (15 noded prism)
2. 2D isoparametric rectangular and triangular elements:
   a) 36200 (4 noded rectangles)
   b) 36210 (8 noded rectangles)
   c) 36100 (3 noded triangles)
   d) 36110 (6 noded triangles)
   Control options PLANE.STRESS and PLANE STRAIN have been tested.
3. 2D Semi-Loof rectangular and triangular elements
   a) 43210 (8 noded rectangles)
   b) 43110 (6 noded triangles)

5.2.3 Convergence Module

Validation of the CONVERGENCE module was investigated by varying parameters and then checking that the program performed as expected. This was done on several of the test jobs. The load increment defined in this module refers to the stored load increment and sometimes it is possible to apply tolerances to the previous increment. Care must be taken when using this feature.

5.2.4 Tolerance Module

Validation of the TOLERANCE module like the convergence module was performed on various test jobs and found to work in conjunction with the CONVERGENCE module.

6. Conclusion

This report has outlined the implementation of the unified constitutive model into the PAFEC Finite Element Program Level 8.1. Several new enhancements have made this implementation capable of solving large cyclic plasticity problems quickly and efficiently. This version has reduced solution times for large finite element models down to a fraction of the computational time that was previously required. Application of this work will enable the determination of the residual stress fields that exist in the F-111C Wing Pivot Fitting after multiple CPLT load cycles. The results of that analysis will provide the required residual stress information to be used as an essential ingredient in the determination of inspection intervals for the RAAF F-111C fleet of aircraft.

7. Acknowledgments

The author wishes to thank the staff at PAFEC Ltd for their help and detailed information on the workings of the Finite Element Code. In addition many thanks go to past and present AMRL staff members have contributed to the overall work program in the areas of structural testing and structural mechanics.

8. References

1) Ramaswarmy, V. G. 'A Constitutive Model for the Inelastic Multiaxial Cyclic Response of a Nickel Based Superalloy René 80', NASA Contractor Report 3998, July 1986.

2) Bridgford, N., 'A Summary of the Bodner-Stouffer Constitutive Model', Aeronautical Research Laboratories, TM-513, DSTO, Melbourne, Aust, May 1989.

3) Paul, J., Bridgford, N. and Stouffer, D., 'Progress Report on Implementation of a Constitutive Model into the PAFEC Finite Element Package', Aeronautical Research Laboratories, Struct-TM-529, DSTO, Melbourne, Aust, June 1990.

4) B. Trippit, J. Tomas, and R. Jones. 'Modelling the inelastic response of materials in ABAQUS using the Ramaswarmy-Stouffer-Bodner constitutive law. In ABAQUS Users' Conference Proceedings, pages 543-555, Newport, Rhode Island, USA, June 1994.

5) Trippit B. and Jones R., 'Recent Australian developments in the engineering applications of constitutive modelling'. In Numerical Implementation & Application of Constitutive Models in the Finite Element Method. ASME International Mechanical Engineering Congress & Exposition, ASME, November 1995.

6) Stouffer, D.C. and Dame, L. T., 'Inelastic Deformation of Metals: Models, Mechanical Properties, and Metallurgy', John Wiley & Sons Inc., New York March 1995.

7) Searl A. and Paul J., 'Characterisation of D6ac Steel using a Unified Constitutive Model', AMRL File No M1/8/982, 1996.

8) PAFEC Ltd. 'Data Preparation User Manual Level 7.1', PAFEC Limited, Strelly Hall, Nottingham, NG8 6PE, United Kingdom.

9) 'Enhancement of the unified constitutive model', Technical Support Services Contract Number 639207 let to MONSAFE, Monash University, Clayton, 3168, Australia.

10) Allan R. B., 'Elastic-plastic analysis of a plate of strain hardening material with a central circular hole - Comparison of experiment with finite element analysis containing the unified constitutive material model', DSTO-TR-0492, February 1997.

11) Allan R. B., 'Elastic-plastic analysis of a plate with a central elongated hole - Comparison of experiment with finite element analysis containing the unified constitutive material model', In press.

12) Hindmarsh A. C., ODEPACK, a systematized collection of ADE solvers. In R. Stepleman et al., editors, Scientific Computing, volume 1, pages 55-64. IMACS/North-Holland Publishing Company, North-Holland, Amsterdam, 1983.

Tables

Table 1: Non PAFEC Subroutine Description.

Subroutine Name	Description
JP1000	Initialisation and data validation.
JP1010	Initialises load scratch modules.
JP1011	Extends incremental solution size.
JP1012	Extracts load case data from TOTAL.TIME module.
JP1100	Load increment control routine. Determines structural time step.
JP1101	Determines % load based on current time for each Load Case.
JP1102	Manipulates incremental and dynamic modules.
JP1103	Determines if current increment is to be stored to the results files.
JP1104	Prints the summary table header line.
JP1105	Interpolates % load to be applied time for each load case.
JP1106	Determines if an elastic solution has been requested at a turning point.
JP1110	Determines whether material Jacobian is to be calculated for use next iteration.
JP1120	Reads/writes backup of non-linear data sets and displacements.
JP3000	Organises variable retrieval and storage, executes the new unified constitutive algorithm. Called by PL3000 and PL3100.
JP3002	Extracts unified control information.
JP3100	Internal loads for thick and thin shell elements.
JP3900	Retrieves stress and strain components for a single Gauss point.
JP3901	Retrieves and Stores state variables for a single Gauss point.
JP3910	Stores stress and strain components for a single Gauss point.
JP3920	Determines minimum material time step for next load step.
JP3921	Updates reference stress, strain and state variables.
BT4800	Calculates equivalent stress, equivalent plastic strain, equivalent elastic strain and equivalent total strain.
BT6000	Controls the calculation of the elasto-plastic modulus matrix.
BT6100	Extracts the elasto-plastic modulus matrix from storage.
JPUTIL	Contains some commonly used routines.
kmodel	Main unified constitutive routine. Called by JP3000.
kkmodel	Performs the call to the integration scheme for each time step.
kdy	Provides derivatives as a function of time for Livermore solver.
kak2	Calculates K2 in the constitutive equations.
keffdvs	Calculates effective stress.
keffeps	Calculates effective strain rates.
kepskk	Calculates the sum of the normal components of the total strain.
kwrkid	Calculates the work.
lsode	Livermore solver.

Table 2: Modified or Re-written PAFEC Subroutines

Subroutine Name	Modifications
R09702	Sets control LFLAGS in common block.
NL1000	Main initialisation control routine.
PL1020	Retrieves modules 228,229,230 and 231 into BASE.
NL1041	Calculates the length of BASE required to store each data set.
PL1040	Creates work space for plasticity solutions.
NL1100	Main load increment initialisation control routine.
NL1101	Prints the load summary table header.
NL1110	Determines the type of solution to perform next iteration.
PL3000	Organisation of variable retrieval and storage prior to and post state determination for 2D and 3D isoparametric elements.
PL3100	Internal loads for thin and thick shell elements. Calls JP3100.
NL1150	Writes last block of the non-linear data.
NL1500	Calculates the residual forces.
NL1600	Tidies up at the end of an incremental solution.
NL1300	Checks structural sub-iteration convergence.
NL1340	Determines if an iterative solution has converged.
NL1360	Print and nodal extrapolation control.
R00411	Stops SS file warning messages.
R70620	Stress averaging routine.
NL0007	Reduces order of integration to 2x2x2.
L36300	Controls all stress/strain printed output.
L36301	Controls all stress/strain output to results files.
PL5000	Calls relevant yield function routine. By-passed.
PL6000	Controls the calculation of the elasto-plastic modulus matrix.
PL6100	Calculates the elasto-plastic modulus matrix.
PL6200	Calculates the elasto-plastic modulus matrix for 2D iso-parametric elements.
NL9200	Main stress/strain routine for 361XX and 362XX series of elements.
NL9220	Calculates and prints extrapolated stress/strain components.
NL9222	Calculates principal stresses and strains.
NL9300	Main stress/strain routine for 371XX and 372XX series of elements.
NL9320	Calculates and prints extrapolated stress/strain components.
NL9322	Calculates principal stresses and strains.
NL9400	Main routine for semi-Loof, 43XXX series of elements.
NL9410	Sets up to write stress/strain data for semi-Loof elements.
NL9412	Writes stress/strain data for semi-Loof elements.
NL9500	Main stress/strain routine for beam, 34XXX series of elements.
NL9320	Calculates and prints extrapolated stress/strain components.
NL9321	Returns stress/strain values at corner integration points.
NEWLIN	Stops new lines from appearing in constitutive output.

Table 3: Summary of Test Jobs[1]

Test #	Data File	Element Type	Loading Condition	Description
1	T3D1	37110	Free Uniaxial Pull	Test the general stress to strain response and volumetric response.
2	T3D2	37110	Constrained Pull	Test the general stress to strain response and volumetric response under different hydrostatic conditions than test 1.
3	T3D3	37100	Shear	Test the general stress to strain response in shear.
4	T3D4	37110	Free Uniaxial Cyclic	Test the cyclic stress to strain response.
5	TPS1	36210	Free Uniaxial Pull	Plane stress. Test the general stress to strain response and volumetric response.
6	TPS2	36210	Constrained Pull	Plane stress. Test the general stress to strain response and volumetric response under different hydrostatic conditions than test 5.
7	TPS3	36200	Shear	Plane stress. Test the general stress to strain response in shear.
8	TPS4	36210	Free Uniaxial Cyclic	Plane stress. Test the cyclic stress to strain response.
9	TPE1	36210	Free Uniaxial Pull	Plane strain. Test the general stress to strain response and volumetric response.
10	TPE2	36210	Constrained Pull	Plane strain. Test the general stress to strain response and volumetric response under different hydrostatic conditions than test 9.
11	TPE3	36200	Shear	Plane strain. Test the general stress to strain response in shear.
12	TSH1	43210	Free Uniaxial Pull	Test the general stress to strain response and volumetric response.
13	TMU1	37110 36210 43210	Free Uniaxial Pull.	Multiple element type tests.

[1] Date files and Result files reside on the AMRL computational server 'bigted' in the directory '/pafec/constitutive/source/validation/runs'.

Figures

```
              NL1000    Create workspace modules or check for restart.
|-------> NL1100    Next increment. Re-init. and print increment summary.
|   |---> NL1110    Next iteration. Check for updated tangent stiffness.
|   |     R03000    Update tangent stiffness and internal loads.
|   |     R3XXXX    Main stiffness calculating routines for a particular Element.
|   |               R37XXX for 3D Isoparmetric Elements.
|   |               R36XXX for 2D Plane Stress and Plane Strain Elements.
|   |               R43XXX for 2D Thin Shell Elements.
|   |     R03001    Clean up after stiffness generation.
|   |     C15500    Calculates new displacement increments.
|   |     NL1140    Form total displacements and update geometry.
|   |     NL1550    Prepare for residual force calculation.
|   |     R09130    Sets up modules prior to stressing.
|   |     R8XXXX    Main stress routines for a particular Element Type.
|   |     R09135    Cyclic symmetry module swapping.
|   |     NL1150    Write last blocks to nonlinear data sets.
|   |     NL1160    Decide which method to use to generate internal forces.
|   |     R03000    Update tangent stiffness and internal loads.
|   |     R3XXXX    Main stiffness calculating routines for a particular Element.
|   |     R03001    Clean up after stiffness generation.
|   |     NL1170    Update any element loads to current geometry.
|   |     R03000    Update tangent stiffness and internal loads.
|   |     R3XXXX    Main stiffness calculating routines for a particular Element.
|   |     R03001    Clean up after stiffness generation.
|   |     NL1180    Check if solution method is iterative.
|   |     NL1500    Calculate the residual forces for this iteration.
|   |     NL1300    Print solution tolerance metrics.
|   |---< NL1130    Switch the ES and FS files if necessary.
|-------< NL1360    Increment has converged, or iteration limit exceeded, print.
          NL1600    End of solution house keeping.
          R70600    Stress averaging performed.
          R70620    Write last block of stress file.
```

Figure 1: Main FORTRAN plasticity routine calls

Initialisation

R09702*			
NL1000*	- PL1020*		
	- JP1000	- JP1012	
		- JP1010	- JP1012
			- JP1011
	- NL1040	- NL1041*	
	- PL1040*		

Load step house keeping

NL1100*	- JP1100	- JP1103	
		- JP1101	- JP1105
		- JP1102	
		- JP1106	
		- JP1120	
	- NL1101*		
	- JP1104		
NL1110*	- JP1110		

Stiffness matrix generation

PL6000*	- BT6000	- BT6100
PL6100*	- BT6100	
PL6200*	- BT6100	

Stress determination

PL3000*	- JP3000	- JP3900		
		- BT4800		
		- JP3002		
		- JP3901		
		- kmodel	- kkmodel	- lsode
				- kepskk
			- kepskk	
			- keffdvs	
			- kkmodel	
		- JP3901		
		- JP3910		
PL3100*	- JP3100	- JP3000	- as above	

Post convergence processing

NL1150*		
NL1500*		
NL1300*	- NL1340*	
NL1360*	- JP3920	- JP3921

NL9XXX series of stressing routines

<u>Miscellaneous</u>

R00411*	L36300*	PL5000*	R70620*
NL0007*	L36301*	BT4800	NEWLIN*

<u>NOTES</u>

* = Original PAFEC subroutine modified.
\- = Subroutine calls this routine.
All subroutines beginning with JP or BT have been written from scratch.

Figure 2: Program Flow Chart of Modified Routines

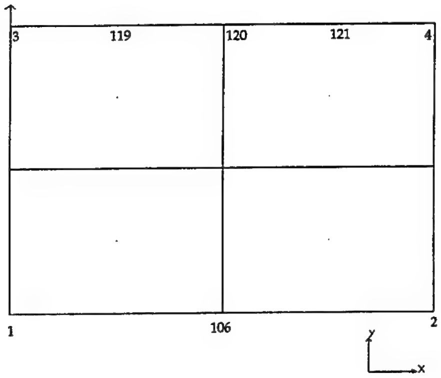


Figure 3a:Test 5 Applied Displacements.

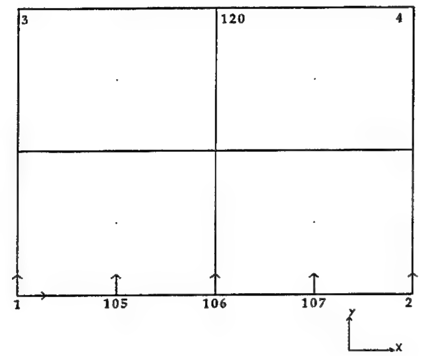


Figure 3b: Test 5 Applied Restraints.

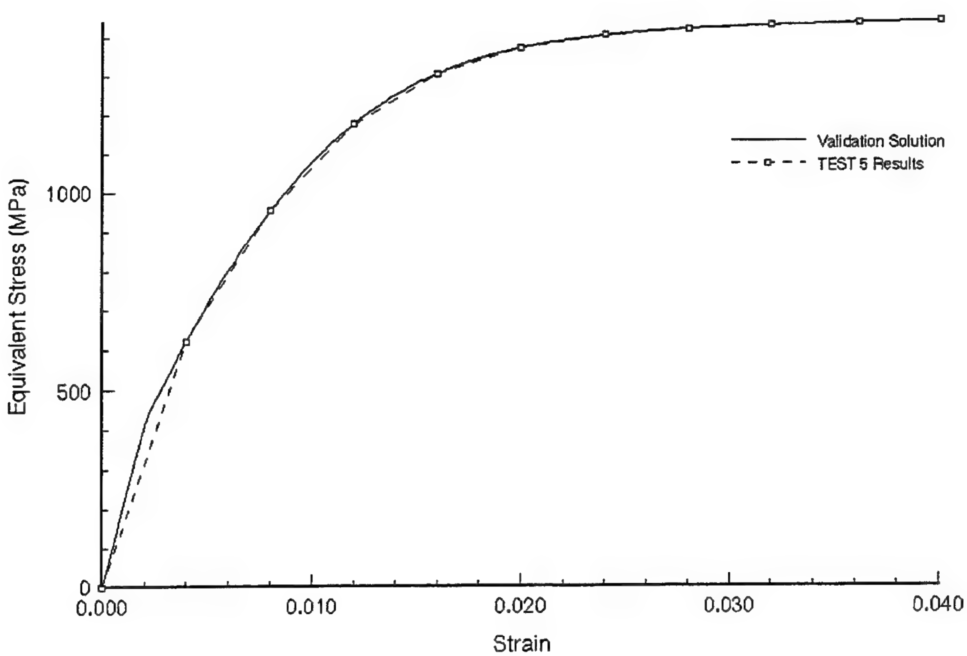


Figure 4: Results for Validation Test 5.

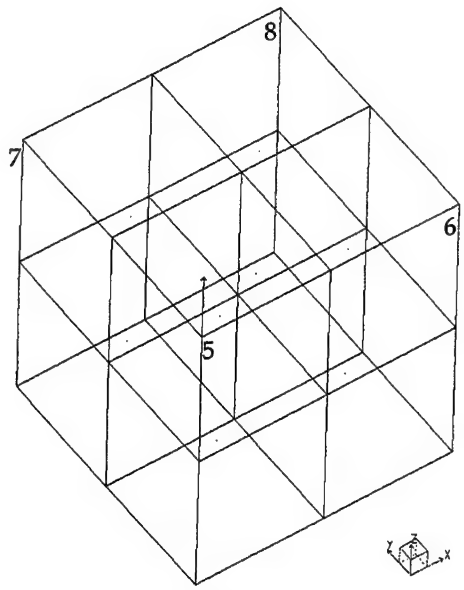


Figure 5a:Test 1 Applied Displacements.

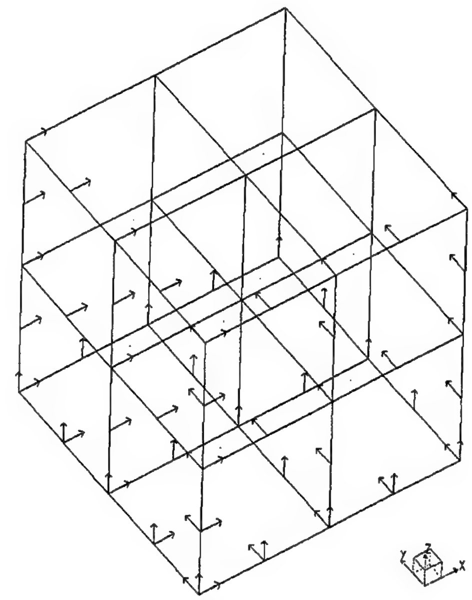


Figure 5b: Test 1 Applied Restraints.

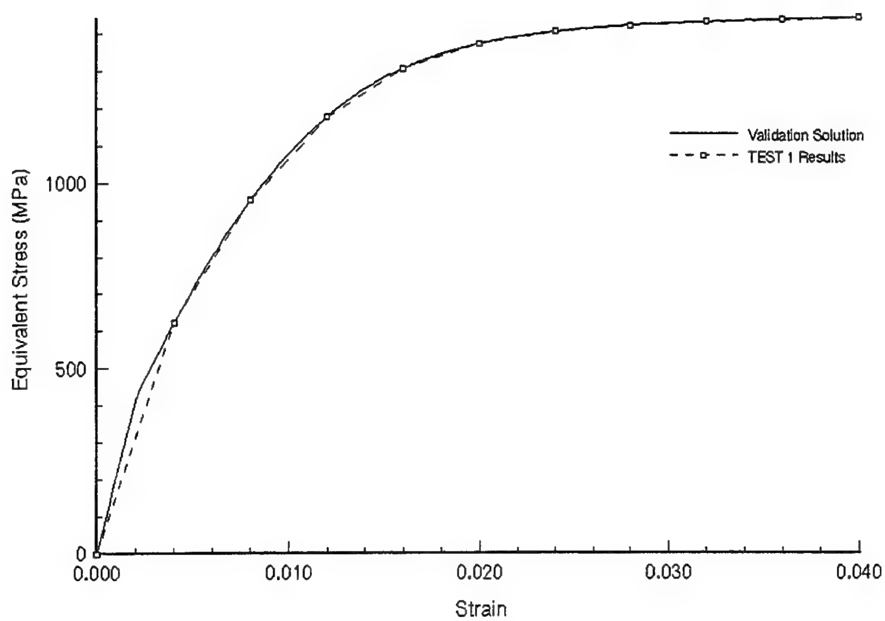


Figure 6: Results for Validation Test 1.

Appendix A

INPUT DATA MODULE DESCRIPTION

A.1: AUTOMATIC.CONTROL

AUTO.CONTROL	TOTAL.TIME.STOP	OUTPUT.PRINT.CONTROL	TABLE.PRINT.CONTROL

NUMBER.OF.LOAD.CYCLES	BLOCK.SIZE	DELTA.STORAGE.TIME.VALUE

RE.FACTOR	EX.FACTOR	RE.ITERATION	EX.ITERATION

INITIAL.DELTA.TIME	ABORT.REDUCTION.TIME.FACTOR	MAXIMUM.TIME.STEP

This module defines the global control parameters required for the unified constitutive solution.

AUTO.CONTROL

Specifies whether automatic time stepping is to be used.

= 1 Automatic control
values in TOTAL.TIME.VERSUS.FORCE.DATA correspond to total time.

= 0 Manual Control
values in TOTAL.TIME.VERSUS.FORCE.DATA correspond to delta time.
THIS OPTION IS RARELY USED AND IS UNTESTED IN THIS VERSION.

Range: 0 and 1
Default = 1

TOTAL.TIME.STOP

Determines the termination time for the solution.
If greater than the maximum referenced time in the TOTAL. TIME.VERSUS.FORCE.DATA module the lower value is used and a warning message is printed in the phase output.

Range: > 0
No Default

OUTPUT.PRINT.CONTROL

This options allows the output written to the phase output file to be controlled or totally stopped.

Value	Meaning
0	No output
1	Iteration convergence information printed.
10	Displacements printed.
100	Gauss point information printed.
1000	Stress/strain data printed
10000	Cumulative reactions printed
2	All of the above.

Combinations of the above values are allowed eg if displacements and stresses were required to be printed then this option would be 1010 ie 10 + 1000.
Default = 0

TABLE.PRINT.CONTROL

Allows a table of loading and time stepping information to be written to the phase output.
= 0 No table write
= 1 Table write

Range: 0 and 1
Default = 1

NUMBER.OF.LOAD.CYCLES

Specifies the number of times the loading history is to be repeated. The loading is defined as one copy of the force time curve, duplicated for each load case, input via the INCREMENTAL and TOTAL.TIME.VERSUS.FORCE. DATA modules.
UNTESTED IN THIS VERSION.

Range: ≥ 1
Default = 1

BLOCK.SIZE

This is the total number of increments that may be stored in a solution run. For large problems this may cause large disk space usage. One block is required per increment stored.

Range: > Maximum number of increments defined in the INCREMENTAL module.
Default = 20

DELTA.STORAGE.TIME.VALUE

This is the time interval for storage of results in output files. Results are also written at the end of each user defined loading point irrespective of the specified value. Results can be stored every increment by specifying zero.

Range: ≥ 0
Default = 0

RE.FACTOR

Specifies the factor by which the time step should be reduced if the iteration count from the previous step is greater or equal to RE.ITERATION.

Range: $0 < RE.FACTOR \leq 1$
Default = 0.85

EX.FACTOR

Specifies the factor by which the time step should be expanded if the iteration count from the previous step is less or equal to EX.ITERATION.

Range: ≥ 1
Default = *1.15*

RE.ITERATION

Specifies the iteration number at which the time step for the next increment will be reduced by the factor RE.FACTOR See Figure A.1

Range: > *EX.ITERATIONS*
Default = *8*

EX.ITERATION

Specifies the iteration number at which the time step for the next increment will be expanded by the factor RE.FACTOR. See Figure A.1

Range: *0* < *EX.ITERATIONS* < *RE.ITERATIONS*
Default = *4*

INITIAL.DELTA.TIME

Specifies the initial time step at the start of run when in automatic control.

Range: > *0*
Default = *2 seconds*

ABORT.REDUCTION.TIME.FACTOR

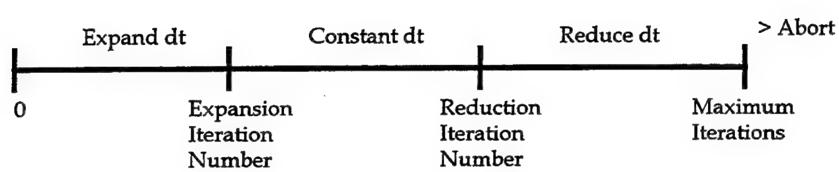
Specifies the factor by which the time step should be reduced and the increment re-attempted in the case of a structural convergence failure, maximum iteration exceedance or a material point convergence failure.

Range: > *0*
Default = *0.5*

MAXIMUM.TIME.STEP

Specifies the maximum time step allowed. This is used to limit the time step from expanding past unrealistic values.

Range: > *0*
Default = *5 seconds*



Line represents number of iterations to convergence for the previous increment.

dt = Time Step

Figure A.1: Structural Time step calculation logic.

A.2: CONSTANTS.NEW.FLOW.LAW

MATERIAL.NUMBER	TYPE.MATERIAL	TEMP

E	NU	D	N	Z0	Z1	M	F1	F2	OMMAX

Defines the state variables properties to be used by the unified constitutive code.

MATERIAL.NUMBER

Property number referenced by the PLASTIC.MATERIAL module, given to the following properties. If the properties are temperature dependent, the MATERIAL.NUMBER is repeated for each temperature where the state variables are known.

No Default

TYPE.MATERIAL

Two forms of the unified constitutive equations currently exist.
= 1 General purpose equations
= 2 Form used for high temperature material Hastelloy X.

Range: 1 and 2
No Default

TEMP

Temperature of current set of constitutive state variables properties.

No Default

E

Young's modulus.

No Default

NU

Poisson's Ratio.

No Default

D

Inelastic flow scale factor.

No Default

N

Inelastic flow strain rate sensitivity parameter.

No Default

Z0

Initial drag stress.

No Default

Z1

Saturated drag stress.

No Default

M	Cyclic hardening parameter. *No Default*
F1	Strain hardening parameter. *No Default*
F2	Yield parameter. *No Default*
OMMAX	Maximum back stress. *No Default*

A.3: TOTAL.TIME.VERSUS.FORCE.DATA

LOAD.CASE	LIST

Time data associated with the INCREMENTAL module resulting in a load-time curve to be applied to the structure.

LOAD.CASE

The number of the load case in which the TOTAL.TIME. VERSUS.FORACE.DATA is to be applied.

7

Default = 1 or 1 + the previous entry

LIST

This is a list of time points that are associated with the INCREMENTAL module. In automatic time stepping mode this list represents the total time to obtain the required load. In manual mode this is a list of delta time for each increment in load step.

No Default

NOTE 1: For multiple load cases a value of 0 is acceptable.

NOTE 2: A negative value infers the use of the elastic modulus at the next increment.

<u>EXAMPLE</u> 1 Load Case

```
TOTAL TIME VERSUS FORCE DATA
LIST
C -----AUTOMATIC MODE -- Total = 30 seconds.
10, 20, 30
C -----MANUAL MODE -- Total = 2.4 seconds.
.5, .5, .3, .2, .1, .1, .05, .05, .1, .2, .3
```

<u>EXAMPLE</u> 2 Load Cases

```
TOTAL TIME VERSUS FORCE DATA
LOAD  LIST
C -----AUTOMATIC MODE -- Total = 60 seconds.
1     10, 20, 30
2      0,  0,  0, 40 50 60
```

A.4: UNIFIED.CONTROL

JACOBIAN.CALCULATE	METHOD.SOLUTION	ORDER.SOLUTION

PERTURBATION.DISTANCE.FACTOR	REEPSI	REOMI	REWRKI	REQEPSI

AEEPSI	AEOMI	AEWRKI	AEQEPSI

OOH.YOUR.STRAIN.IS.NOT.LINEAR	ELASTIC.JACOBIAN.FACTOR

This module specifies local control parameters for the constitutive model including solution schemes, integration tolerances and factors controlling material Jacobian determination.

JACOBIAN.CALCULATE

Specifies the method by which the material Jacobian will be calculated.
0 = Assumed elastic.
2 = Numerically determined by divided differences. Perturbation

Range: 0 and 2
Default = 0

METHOD.SOLUTION

Specifies the scheme for the constitutive integration. Refer to Hindmarsh [12] for full description.
= 10 Non stiff solution.
= 21 Stiff solution requiring full Jacobian matrix df/dy.
= 22 Backward Differentiation Formula (BDF) using numerical Newton requiring full Jacobian matrix df/dy.
= 24 Stiff solution requiring banded Jacobian matrix df/dy.
= 25 Backward Differentiation Formula (BDF) using numerical Newton requiring banded Jacobian matrix df/dy.

Range: 10, 21, 22, 24 and 25
Default = 22
WARNING: This option is intended for experienced users only.

ORDER.SOLUTION

Specifies the maximum order for the constitutive integration. Refer to Hindmarsh [12] for full description.

Range: Range 1-5 for BDF method and 1-12 for Adam's method.
Default = 5

WARNING: This option is intended for experienced users only.

PERTURBATION.DISTANCE.FACTOR

This field specifies the perturbation factor which controls the perturbation distance (strain) for numerically determined

material Jacobians. The perturbation distance is equal to the perturbation factor multiplied by an estimate of the error on the strain. The perturbation factor approximately controls the number of significant figures in the Jacobian. For example F_{pert}=1000 or 10^3 will result in approximately 3 significant figures in the Jacobian. Refer to [9] of this report for a further description.

Range: > 0
Default = 1000

NOTE: This control is coupled with integration tolerances. An increase in perturbation factor will generally require a similar decrease in integration tolerances.

REEPSI Specifies the allowable local relative error on the integration of inelastic strain.

Range: > 0
Default = 5E-8

REOMI Specifies the allowable local relative error on the integration of inelastic back stress.

Range: > 0
Default = 5E-8

REWRKI Specifies the allowable local relative error on the integration of inelastic work.

Range: > 0
Default = 5E-8

REQEPSI Specifies the allowable local relative error on the integration of equivalent inelastic strain.

Range: > 0
Default = 5E-8

AEEPSI Specifies the allowable local absolute error on the integration of inelastic strain.

Range: > 0
Default = 1E-11

AEOMI Specifies the allowable local absolute error on the integration of inelastic back stress.

Range: > 0
Default = 1E-6

AEWRKI

Specifies the allowable local absolute error on the integration of inelastic work.

Range: > 0
Default = 1E-12

AEQEPSI

Specifies the allowable local absolute error on the integration of equivalent inelastic strain.

Range: > 0
Default = 1E-11

OOH.YOUR.STRAIN.IS.NOT.LINEAR

Specifies an allowable nonlinear strain history factor. If the strain history becomes more nonlinear than the specified factor a warning message is printed to the screen and to the phase 9 output. The warning messages are printed when the maximum change in strain rate of this step normalised by the maximum strain rate of the previous step is greater than the specified factor. For example, when the factor is set to 0.4 warnings will be issued when the change in strain rate is greater than 40%. This option is useful for assessing the accuracy of time discretization as the constitutive implementation assumes the strain history is linear between solution points. If warning messages occur the solution should be rerun with decreased step sizes. This option can be disabled by setting the factor to zero.

Range: ≥ 0
Default = 0 (disabled)
Recommended 0.4 to 0.6

NOTE: This field controls only the output of warning messages and does not affect the solution in any other way.

ELASTIC.JACOBIAN.FACTOR

Specifies a measure of inelasticity below which the Jacobian is assumed to be equal to the elastic stiffness matrix. Numerical perturbation to calculate the material Jacobian is only performed when the inelastic strain increment normalised by the total strain increment (at the material point concerned) is greater than the specified factor. The correct use of this factor greatly increases the performance of near elastic material points.

Range: > 0
Default = 0.05

Appendix B

B.1: PAFEC Data Module Definitions For The Module Dictionary

CONSTANTS.NEW.FLOW.LAW	MODULE 228	Defaults	
MATERIAL.NUMBER		-11	1
TYPE.MATERIAL		-11	1
TEMP		-11	0
E		0	0
NU		0	0
D		0	0
N		0	0
Z0		0	0
Z1		0	0
M		0	0
F1		0	0
F2		0	0
OMMAX		0	0

AUTOMATIC.CONTROL	MODULE 229	Defaults	
AUTO.CONTROL		1	1
TOTAL.TIME.STOP		0	0
OUTPUT.PRINT.CONTROL		0	1
TABLE.PRINT.CONTROL		1	1
NUMBER.OF.LOAD.CYCLES		1	1
BLOCK.SIZE		20	1
DELTA.STORAGE.TIME.VALUE		0	0
RE.FACTOR		0	0
EX.FACTOR		0	0
RE.ITERATION		8	1
EX.ITERATION		4	1
INITIAL.DELTA.TIME		0	0
ABORT.REDUCTION.TIME.FACTOR		0	0
MAXIMUM.TIME.STEP		0	0

TOTAL.TIME.VERSUS.FORCE.DATA	MODULE 230	Defaults	
LOAD.CASE		-81	1
LIST		-40	0

UNIFIED.CONTROL	MODULE 231	Defaults	
JACOBIAN.CALCULATE		0	1
METHOD.SOLUTION		22	1
ORDER.SOLUTION		5	1
PERTURBATION.DISTANCE.FACTOR		0	0
REEPSI		0	0
REOMI		0	0
REWRKI		0	0
REQEPSI		0	0
AEEPSI		0	0
AEOMI		0	0
AEWRKI		0	0
AEQEPSI		0	0
OOH.YOUR.STRAIN.IS.NOT.LINEAR		0	0
ELASTIC.JACOBIAN.FACTOR		0	0

B.2: Control Flags Common Block Definitions

Subroutine R09702 sets up the common block LFLAGS, which contains a number of logical flags which describe the current analysis. The following is a detailed list of the common block LFLAGS revised from Reference 3.

NUMBER	DESCRIPTION
1	INCREMENTAL SOLUTION FLAG.
2	GEOMETRIC NON-LINEARITY FLAG.
3	LARGE DISPLACEMENT ANALYSIS.
4	LINEAR BUCKLING.
5	FREQUENCIES OF STRESSED STRUCTURES.
6	PLASTICITY ANALYSIS.
7	CREEP ANALYSIS.
8	MATERIAL NON-LINEARITY.
9	SEISMIC ANALYSIS.
10	STRESS ELEMENTS IN FRONT ORDER.
11	STRESSES AT INTEGRATION POINTS.
12	INCREMENTAL STRESS/STRAIN SOLUTION.
13	RECALCULATE ELEMENT LOADS
↓	
↓	THE REMAINING FLAGS ARE NOT USED BY PAFEC LEVEL 8.1.
↓	
↓	UNIFIED CONSTITUTIVE MODEL FLAGS DESCRIPTION.
↓	
30	SUB-ITERATION EXCEEDENCE FLAG.
31	AUTOMATIC (TRUE) OR MANUAL (FALSE) FLAG.
32	OUTPUT PRINT CONTROL FLAG.
33	TABLE PRINT CONTROL FLAG.
34	UNIFIED CONSTITUTIVE MAIN CONTROL FLAG.
35	TURNING POINT INDICATOR FLAG.
36	LOAD INCREMENT STORAGE FLAG.
37	ITERATION PRINT FLAG.
38	DISPLACEMENT PRINT FLAG.
39	GAUSS POINT PRINT FLAG.
40	STRESS PRINT FLAG.
41	CUMULATIVE REACTION PRINT FLAG.
42	MAXIMUM ITERATION NON CONVERGENCE FLAG.
43	PREVIOUS INCREMENT TURNING POINT FLAG.
44	ELASTIC UNLOAD FLAG.
45 → 50	PRESENTLY NOT USED

B.3: Information on Non-linear Data Sets

The following information provides a description of the variables held in the non-linear data set `NLSET'.

POSITION	CODE	DESCRIPTION
1	*	REFERENCE STRESS VALUES.
2	*	STRESSES.
3	*	REFERENCE TOTAL STRAINS.
4	*	TOTAL STRAINS.
5	*	REFERENCE ELASTIC STRAINS.
6	*	ELASTIC STRAINS.
7	*	REFERENCE PLASTIC STRAINS.
8	*	PLASTIC STRAINS.
9	O	REFERENCE CREEP STRAINS.
10	O	CREEP STRAINS.
11	*	REFERENCE PLASTIC STATE VARIABLES.
12	*	REFERENCE PLASTIC STATE VARIABLE INCREMENTS.
13	*	EQUIVALENT STRESSES AND STATE INDICATORS.
14	O	LOAD HISTORY FLAGS.
15	O	ROTATION MATRIX AND LEFT STRETCH TENSOR.
16 ↓ 18		PRESENTLY NOT USED
19	*	REFERENCE JACOBIAN.
20	*	JACOBIAN.
21	*	REFERENCE TIME STEP DATA.
22	*	TIME STEP DATA.
23	*	REFERENCE PLASTIC BACK STRESS.
24	*	PLASTIC BACK STRESS.
25	*	REFERENCE SCALAR VARIABLES.
26	*	SCALAR VARIABLES.
27 ↓ 50		PRESENTLY NOT USED

<u>Code</u>

* Data sets used in the unified constitutive model code.

O Data sets NOT used in current analysis.

B.4: Created Scratch Module Information.

Several scratch modules are created at initialisation time to hold single and double precision data. The identifier numbers are stored in common block NEWFLOW and within scratch module JPSR1 itself. The common block NEWFLOW has been declared as:
COMMON /NEWFLOW/ JPSR1,JPSR2,JPSR3,JPSR4,JPSR5,JPSR6(10,2)

A description of the contents of each scratch module is provided for reference below:

MODULE *JPSR1* (Created JP1000)

This module holds global information required for the implementation of the unified constitutive model code. A description is provided below:

POSITION	VALUE	DESCRIPTION
1	0.0	Total time at any point in solution.
2	TINIT	Initial delta time step. Later the current global delta time.
3	TOLTM	Termination time of solution.
4	0.0	Copy of location 2 -. Used when sub-iteration convergence is not achieved.
5	IBLOK	Block size to increase allowed load increments.
6	MODTI	Module number holding variables for Load-Time interpolation for each load case, (Setup JP1010, used IN JP1101, JP1102)
7	NSEG	Global time segment pointer. (Used JP1010)
8	MODEL	Module number holding elastic unload time information. 0 then no elastic unloads exist. (Setup JP1010, used JP1106)
9		UNUSED
10	CONVIT	Number of iterations to converge. (Setup NL1100, used JP1100)
11	TURTIM	Storage of next turning point time. (Setup JP1101, used JP3920)
12	ABORFAC	Abort reduction factor. (USED JP1100)
13	AMAXTIM	Maximum time step allowed (used JP1100)
14	TURDELT	Delta time prior to a turning point. Used to recover delta time in JP1100.
15	ASTOR	Delta storage time value. Controls when an increment is written to disk.
16	CTIME	Current print total time. (used JP1103)
17		UNUSED
18		UNUSED
19		UNUSED
20		UNUSED

MODULE *JPSR2* (Created JP1010, Used JP1101,JP1105)

This module holds percentage load versus time values for each load case. This module is created from the expanded incremental module and does not exist in manual control. The created module is printed in the phase 9 output file so that the load-time data may be checked by the user.

MODULE *JPSR3*

This module is currently not used.

MODULE *JPSR4* (Created and Used JP1120)

This module is used to store the previous increments cumulative displacements. This module is recovered when sub-iteration convergence has not been achieved.

MODULE *JPSR5* (Created JP1000, Used JP1105, JP3000, JP3002)

This modules holds the double precision control data required by *kmodel*. This module is a double precision copy of most items in the UNIFIED.CONTROL module 231.

MODULE *JPSR6*(10,2) (Created JP1000, Used JP3000)

This array holds the double precision material data module number in location 1 and the material number in location 2. The module number referenced is a double precision copy of the CONSTANTS.NEW.FLOW.LAW module 228 less the material number for each material number. Each module specifically holds all the state variables associated with each temperature for a particular material number. A maximum of 10 different material numbers are catered for.

MODULE *MODTI* (Created JP1010, Used JP1101, JP1102)

Module number holding variables for Load-Time interpolation for each applied load case. Each line of the module contains the items described below:

ISEG	=	segment in load-time curve.
TURTIM	=	Next turning point time for load curve.
ITURN	=	Flag to indicate that this curve is at a turning point.
ACTIVE	=	Flag to indicate that this load case is active.
ALOAD	=	Actual extrapolated load value at the current time.
4 Variables	=	Used to recover load and accumulated load when non-convergence recovery is required.

MODULE *MODEL* (Created JP1010, Used JP1106)

Module number holding elastic unload time information generated by negative values of time in the TOTAL.TIME.VERSUS.FORCE.DATA module 230. If no elastic unloads exist then this module number is initialised to zero if it does not exist.

DISTRIBUTION LIST

Implementation of a Unified Constitutive Model into the PAFEC Finite Element Package: Final Report
J. Paul

AUSTRALIA

1. DEFENCE ORGANISATION

a. Task Sponsor DTA-LC

b. S&T Program

Chief Defence Scientist
FAS Science Policy } shared copy
AS Science Corporate Management
Director General Science Policy Development
Counsellor Defence Science, London (Doc Data Sheet)
Counsellor Defence Science, Washington (Doc Data Sheet)
Scientific Adviser to MRDC Thailand (Doc Data Sheet)
Director General Scientific Advisers and Trials/Scientific Adviser Policy and Command (shared copy)
Navy Scientific Adviser (Doc Data Sheet and distribution list only)
Scientific Adviser - Army (Doc Data Sheet and distribution list only)
Air Force Scientific Adviser
Director Trials

Aeronautical and Maritime Research Laboratory
Director

Chief of Airframes and Engines Division
F. Rose
K. Watters
A. Searl
J. Paul (5 copies)
C. H. Wang
R. Allan

DSTO Library
Library Fishermens Bend
Library Maribyrnong
Library Salisbury (2 copies)
Australian Archives
Library, MOD, Pyrmont (Doc Data sheet only)

c. Forces Executive
Director General Force Development (Sea) (Doc Data Sheet only)
Director General Force Development (Land) (Doc Data Sheet only)

e. Army
ABCA Office, G-1-34, Russell Offices, Canberra (4 copies)

f. **Air Force**
OIC ASI
ASI2A

g. **S&I Program**
Defence Intelligence Organisation
Library, Defence Signals Directorate (Doc Data Sheet only)

i. **B&M Program (libraries)**
OIC TRS, Defence Regional Library, Canberra
Officer in Charge, Document Exchange Centre (DEC), 1 copy
*US Defence Technical Information Center, 2 copies
*UK Defence Research Information Centre, 2 copies
*Canada Defence Scientific Information Service, 1 copy
*NZ Defence Information Centre, 1 copy
National Library of Australia, 1 copy

2. **UNIVERSITIES AND COLLEGES**

Australian Defence Force Academy
  Library
  Head of Aerospace and Mechanical Engineering
Deakin University, Serials Section (M list), Deakin University Library, Geelong, 3217
Senior Librarian, Hargrave Library, Monash University
Librarian, Flinders University

3. **OTHER ORGANISATIONS**

NASA (Canberra)
AGPS
Mr. J. Platt, PAFEC Ltd., 39 Nottingham Road, Stapleford, Nottingham,
  NG9 8AD, United Kingdom.

OUTSIDE AUSTRALIA

4. **ABSTRACTING AND INFORMATION ORGANISATIONS**
INSPEC: Acquisitions Section Institution of Electrical Engineers
Library, Chemical Abstracts Reference Service
Engineering Societies Library, US
Materials Information, Cambridge Scientific Abstracts, US
Documents Librarian, The Center for Research Libraries, US

5. **INFORMATION EXCHANGE AGREEMENT PARTNERS**
Acquisitions Unit, Science Reference and Information Service, UK
Library - Exchange Desk, National Institute of Standards and Technology, US
National Aerospace Laboratory, Japan
National Aerospace Laboratory, Netherlands

SPARES (10 copies)

Total number of copies: 66

DEFENCE SCIENCE AND TECHNOLOGY ORGANISATION
DOCUMENT CONTROL DATA

1. PRIVACY MARKING/CAVEAT (OF DOCUMENT)

2. TITLE Implementation of a Unified Constitutive Model into the PAFEC Finite Element Package: Final Report	3. SECURITY CLASSIFICATION (FOR UNCLASSIFIED REPORTS THAT ARE LIMITED RELEASE USE (L) NEXT TO DOCUMENT CLASSIFICATION) Document (U) Title (U) Abstract (U)
4. AUTHOR(S) J. Paul	5. CORPORATE AUTHOR Aeronautical and Maritime Research Laboratory PO Box 4331 Melbourne Vic 3001

6a. DSTO NUMBER	6b. AR NUMBER	6c. TYPE OF REPORT	7. DOCUMENT DATE
DSTO-TR-0529	AR-010-220	Technical Report	May 1997

8. FILE NUMBER	9. TASK NUMBER	10. TASK SPONSOR	11. NO. OF PAGES	12. NO. OF REFERENCES
M1/8/992	AIR96/102	DTA-LC	35	12

13. DOWNGRADING/DELIMITING INSTRUCTIONS	14. RELEASE AUTHORITY
None	Chief, Airframes and Engines Division

15. SECONDARY RELEASE STATEMENT OF THIS DOCUMENT

Approved for public release

OVERSEAS ENQUIRIES OUTSIDE STATED LIMITATIONS SHOULD BE REFERRED THROUGH DOCUMENT EXCHANGE CENTRE, DIS NETWORK OFFICE, DEPT OF DEFENCE, CAMPBELL PARK OFFICES, CANBERRA ACT 2600

16. DELIBERATE ANNOUNCEMENT

No Limitations

17. CASUAL ANNOUNCEMENT Yes

18. DEFTEST DESCRIPTORS

finite element analysis, plastic deformation, stress relaxation, F-111C aircraft

19. ABSTRACT

A unified constitutive material model was implemented in to the PAFEC Finite Element Package Level 8.1. The modifications allow the PAFEC Package to perform time dependent plasticity. This includes the calculation of the updated material Jacobian used in the formulation of the global stiffness matrix. Automated time stepping, reduced storage requirements and structural convergence features were also included in the code. A series of test analyses addressing different aspects such as element types and loading conditions were performed and shown to agree with expected results.

TECHNICAL REPORT DSTO-TR-0529 AR-010-220 MAY 1997

DSTO
AUSTRALIA

AERONAUTICAL AND MARITIME RESEARCH LABORATORY
GPO BOX 4331 MELBOURNE VICTORIA 3001
AUSTRALIA, TELEPHONE (03) 9626 7000